Laminar, Two-dimensional Flow Past Cylinder

1   Introduction

This case provides a description for two-dimensional flow past cylinder case with constant properties.

2   Domain

The two-dimensional geometry for the midterm assignment is captured in Figure 1 where a hybrid meshed domain is noted.

The top and bottom surfaces are slip (or zero tangential stress implemented as a "symmetry" boundary condition; the left boundary represents an inflow condition specified velocity, $u_x = U^\infty = 1$. The right boundary condition is open, with a specified static pressure. The cylinder wall boundary condition is no-slip and specified to be zero. Flow past a rotating cylinder can be modeled by adding a rotational slip velocity. Properties for density and viscosity are 1.2 and 0.008, yielding a Reynolds number of 150.

3   Theory

The variable-density low-Mach equation set is defined by the continuity and momentum equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_j}{\partial x_j} = 0. \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_j u_i}{\partial x_j} - \frac{\partial \sigma_{ij}}{\partial x_j} = 0. \tag{2}$$

In the above equation, ρ is the fluid density and u_j is the fluid velocity. The stress tensor is provided by

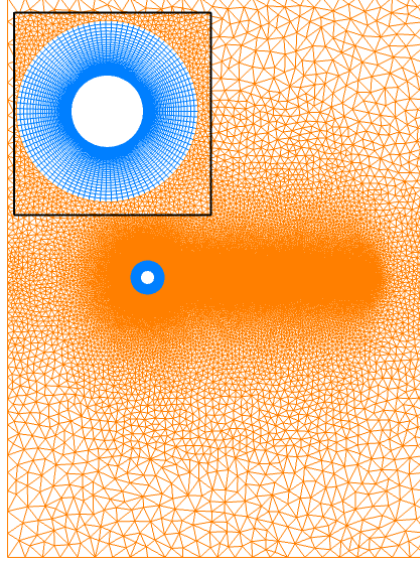$$\sigma_{ij} = 2\mu S_{ij}^* - P\delta_{ij}, \tag{3}$$

Figure 1: Two-dimensional flow past cylinder hybrid geometry (Quad4 and Tri3) in which the diameter of the cylinder, D, is unity.

where the traceless rate-of-strain tensor is defined as

$$S_{ij}^* = S_{ij} - \frac{1}{3}\delta_{ij}S_{kk} = S_{ij} - \frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}.$$

In a low-Mach flow, the above pressure, P, is the perturbation about the thermodynamic pressure, P^{th}.

Introduction

The flow of a liquid or gas past a circular cylinder is ubiquitious in engineering. Examples include the flow of a river current past a bridge support column, or wind blowing past a wind turbine tower or cylindrical water storage tank. This flow is representative of a broad class of flows called "bluff-body" flows. In a bluff body flow, the oncoming fluid at first follows the contours of the body, then at some point separates from the body and creates a wake of relatively low-momentum fluid behind the body. This wake is often of prime importance in engineering of bluff-body systems. It is responsible for creation of a large drag force (force opposing the fluid motion), and can also be responsible for creation of forces on the body that fluctuate in time (also called unsteady forces). The unsteady forces can be oriented both along the wind axis (drag) or perpendicular to the wind axis (lift). Unsteady wakes can also be a source of sound; examples include the wake of aircraft landing gear or "singing" power cables in a strong breeze.

The behavior of a bluff-body flow depends on a non-dimensional number called the Reynolds number. The Reynolds number is defined as:

$$Re \equiv \frac{\rho \, U_\infty \, L}{\mu}. \tag{4}$$

Here, ρ is the density of the fluid, U_∞ is the oncoming fluid velocity relative to the body, or "free-stream" velocity, L is a characteristic length scale associated with the size of the body, and μ is the fluid viscosity. For a circular cylinder, L is usually taken to be the diameter of the cylinder, D. The Reynolds number can be thought of as the ratio of inertial forces to viscous forces in a flow. An example of a low-Reynolds number flow is a marble dropped in a jar of honey. An example of a high-Reynolds number flow is a commercial airliner flying at cruise. For flows where the fluid velocity is much smaller than the speed of sound, the Reynolds number is a similarity parameter; that means any two flows with the same geometry and same Reynolds number behave exactly the same.

Now, back to the bluff-body wake. The behavior of this flow can be dramatically different depending on the Reynolds number. Take the circular cylinder. For very low Reynolds number ($Re < 50$), the flow is steady; in other words, the fluid velocity at any point in space near the cylinder does not vary in time. The wake consists of two symmetric recirculation regions downstream of the cylinder, as shown in Figure 2. As the Reynolds number increases (think of the fluid speeding up as the cylinder size and fluid properties are kept the same), the wake behind the cylinder becomes unsteady. Pockets of swirling fluid, called vortices (plural of vortex), are shed from the cylinder and are swept downstream. For $50 < Re < 200$ this wake remains two-dimensional. Vortices are shed from either side of the cylinder in an alternating pattern called a "vortex street," as shown in Figure 3. This shedding phenomenon causes unsteady lift and drag forces to act on the cylinder at a characteristic frequency, f, called the "shedding frequency." It is convenient to non-dimensionalize this frequency as follows:

$$St \equiv \frac{f \, D}{U_\infty} \tag{5}$$

St is called the Strouhal number, and is a function of the Reynolds number.

As the Reynolds number continues to increase, more complex flow behavior is observed. We will not concern ourselves with the higher Reynolds number regimes, but the interested reader is encouraged to investigate this further on their own.

Quantities of Interest

In any engineering analysis, we must define the relevant Quantities of Interest (abbreviated QoI's), which are the analysis outputs that we are interested in predicting. For the circular cylinder flow, we will define several:

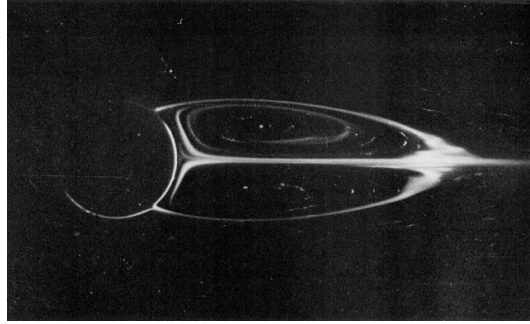- Time-averaged drag coefficient, $\overline{C_d}$

Figure 2: Flow past a circular cylinder at $Re = 41$, from a flow visualization experiment. The flow is from left to right. From Van Dyke, *An Album of Fluid Motion*, 1982.
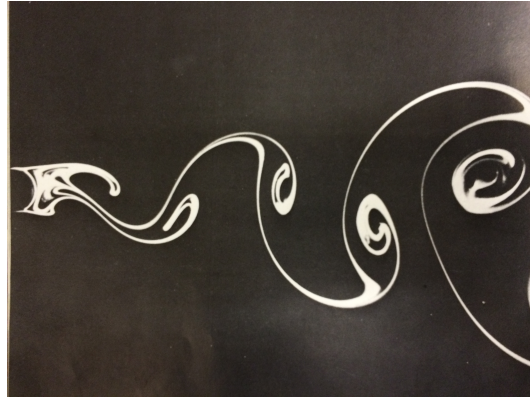

Figure 3: Flow past a circular cylinder at $Re = 140$, from a flow visualization experiment. The flow is from left to right. From Van Dyke, *An Album of Fluid Motion*, 1982.

- Strouhal number, St

- Root-mean-square of the lift coefficient fluctuation, C_l'

We defined the Strouhal number earlier. The drag coefficient is defined as:

$$C_d \equiv \frac{F_d}{1/2\rho U_\infty^2 D}. \tag{6}$$

F_d is the drag force of the fluid acting on the cylinder. This force is comprised of two components: the pressure drag, resulting from the distribution of fluid pressure along the surface of the cylinder; and the viscous drag, resulting from friction as the fluid flows past the surface.

4

Similarly, the lift coefficient is defined as:

$$C_l \equiv \frac{F_l}{1/2\rho U_\infty^2 D}. \tag{7}$$

F_l is the lift force, or force of the fluid acting in a direction perpendicular to the oncoming flow.

A sample passive scalar mixture fraction Nalu result for the $Re = 150$ is shown in Figure 4.
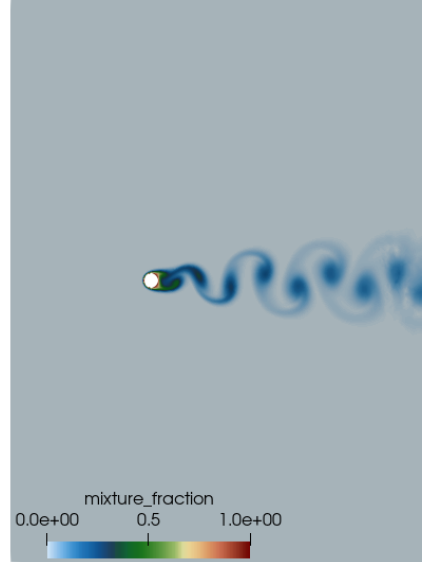
4 Discussion Points

Figure 4: Two-dimensional passive scalar representation for a $Re = 150$ case in which a Schmidt number of 0.9 is specified to compute the diffusive flux coefficient.

- Compare timing between EBVC and CVFEM. Comments on speed?

- Modify the Reynolds number 10x smaller, 10x larger and report general findings. As you change the Reynolds number, make sure that you adjust the initial time step specification such that the first Courant number reported in the log file is not greater than 2.0.

- Explore Peclet blending for both velocity and mixture fraction.

- Explore laminar Schmidt number (nominal value of 0.9), 10x smaller; 10x larger.

- Explore various quantities of interests (QoIs), including: mean and instant drag coefficient; mean and instant lift coefficient.

  Note: *vortexStreetCVFEM_Quad4.dat* that provides forces and moments on the cylinder surface. For QoIs above, at the nominal Reynolds nominal Reynolds number, what is the effect of: a) Mesh refinement; see: *examples/street/circular_cylinder_medium.exo*, b) First-order time vs second-order time: *second_order_accuracy* : *yes/no*, c) Changing the number of nonlinear iterations from 2 to 5, and 10: (note the convergence within each step): *max_iterations* : $2, 4, 8$, d) Linear solver tolerance: tolerance: 1e-5, e) Usage of limiters and the effect

- (Optional) Compute a FFT for either mixture fraction or velocity at select locations/points in the domain.

- (Optional) For CVFEM deactivate upwind blending via the designation of the tanh, *peclet_function_tanh_transition* to something large, i.e. 1e6. Activate nonlinear stabilization operator for mixture fraction by adding the following option:

  element_source_terms : *mixture_fraction* : $[NSO\_2ND|NSO\_4TH]$